

LA-UR-00- 5502

Approved for public release;
distribution is unlimited.

Title: COMPUTATIONAL FLUID DYNAMIC SIMULATIONS OF
INSULATION-DEBRIS TRANSPORT EXPERIMENTS

Author(s): Kyle Ross, OMICRON
Bruce C. Letellier, TSA-11

Submitted to: 28th Annual NRC Water Reactor Safety Meeting
October 23-24, 2000
Bethesda, MD

LOS ALAMOS NATIONAL LABORATORY



3 9338 00789 0451

Los Alamos

NATIONAL LABORATORY



Los Alamos National Laboratory, an affirmative action/equal opportunity employer, is operated by the University of California for the U.S. Department of Energy under contract W-7405-ENG-36. By acceptance of this article, the publisher recognizes that the U.S. Government retains a nonexclusive, royalty-free license to publish or reproduce the published form of this contribution, or to allow others to do so, for U.S. Government purposes. Los Alamos National Laboratory requests that the publisher identify this article as work performed under the auspices of the U.S. Department of Energy. Los Alamos National Laboratory strongly supports academic freedom and a researcher's right to publish; as an institution, however, the Laboratory does not endorse the viewpoint of a publication or guarantee its technical correctness.

COMPUTATIONAL FLUID DYNAMIC SIMULATIONS OF INSULATION-DEBRIS TRANSPORT EXPERIMENTS

Kyle Ross
Omicron Safety and Risk Technologies, Inc.

Bruce Letellier¹
Probabilistic Risk and Hazard Analysis Group
Technology and Safety Assessment Division
Los Alamos National Laboratory

SCOPE

Computational fluid dynamic (CFD) simulations have been performed of water flow through the flume and tank experiments built at the University of New Mexico (UNM) to investigate the transportability of various types of insulation debris that might be generated in a loss-of-coolant accident (LOCA) at a pressurized-water-reactor (PWR) nuclear power plant. The goal of the calculations was to provide insight into how flow patterns and velocities develop in the flume and tank under various configurations of volumetric flow rate and geometric obstacles. Details of the flume CFD calculations are discussed here, and sample graphical results for both the flume and tank are presented.

FLOW-3D

The CFD calculations were performed using the FLOW-3D² computer program. FLOW-3D is a general-purpose software package for modeling the dynamic behavior of liquids and gases that are influenced by a wide variety of physical processes. The program is based on the fundamental laws of mass, momentum, and energy conservation, and it has been constructed to treat time-dependent multi-dimensional problems. FLOW-3D is applicable to many industrial flow processes.

FLUME MODEL

A 20-ft-long water flume was constructed at UNM to study the transport characteristics of various types of insulation debris under well-controlled uniform flows (see photos). Different aspects of the corresponding FLOW-3D model are described below.

Computational Domain

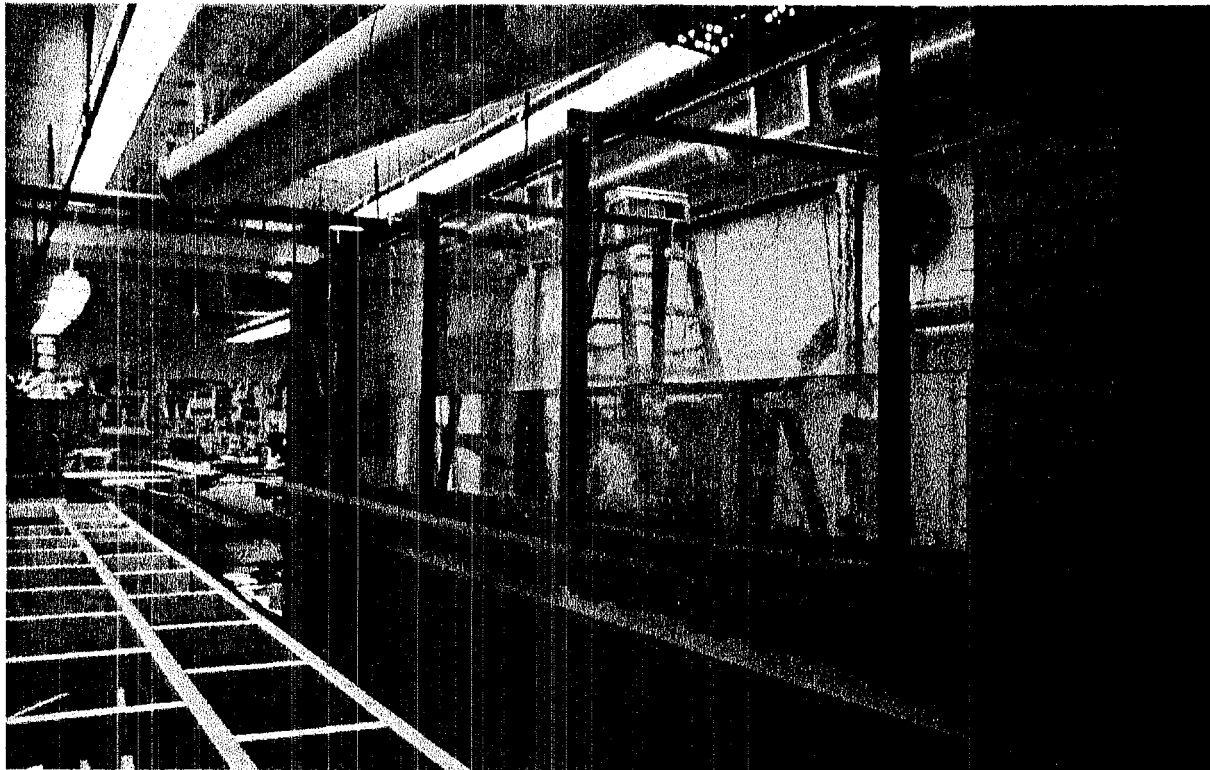
A computational domain was defined with extents to match the physical dimensions of the UNM flume. The flume is 20 ft long, 3 ft wide, and 4 ft high. FLOW-3D grids normally are constructed in rectangular coordinates (cylindrical coordinates are an option), and the cells of the flume model are rectilinear in shape. The typical spatial resolutions in this model are approximately 3 in. in each dimension. Cells along the bottom and sides of the model were defined to be thinner than the typical cell to better resolve velocities at the walls. These cells are 1 in. thick in their dimension normal to the wall.

Internal Structure

The UNM flume has a bank of evaporative-cooler pads and a grid of flow-straightening plates just downstream of where water is introduced to smooth the flow before it enters the test region. The pads (diffuser) and plates (straighteners) are represented in the CFD model with a three-dimensional porous obstacle and two-dimensional baffles, respectively.

¹Bruce Letellier, P.O. Box 1663, MS K557, Los Alamos National Laboratory, Los Alamos, NM 87545, bcl@lanl.gov, (505)665-5188.

²Flow Science, Inc., Los Alamos, New Mexico.



The porous obstacle representing the diffuser is 6 in. thick; it begins 18 in. downstream from the inlet end of the flume and extends across the full width and height of the flume. This is very similar to the size and placement of the actual diffuser. The flow-resistance characteristics of the model diffuser have been defined to give a 4-in.-H₂O pressure drop at a 0.25-ft/s velocity in the flume test section, consistent with measurements made in the UNM flume. The flow-resistance traits of the porous obstacle are defined homogeneously, i.e., equivalently in all directions. This is thought to be consistent with the characteristics of the actual diffuser.

The baffles representing flow straighteners are oriented in the model just as they are in the flume. There are horizontal and vertical baffles that intersect each other to form 3-in.-square horizontal flow passages. Consistent with the flume, the baffles are 1 ft long with their leading edges 2 ft from the inlet end of the flume. The physical straighteners are made of thin sheet steel. FLOW-3D baffles are two-dimensional and therefore have no thickness.

The UNM flume has a small angle bracket attached to the floor to keep the straighteners in place. The downstream end of the straighteners rests against this bracket. The bracket protrudes from the floor approximately ½ in. and spans the width of the flume. This bracket is represented in the FLOW-3D model by a solid object ½ in. tall by ½ in. wide by 3 ft long.

A screen is positioned at the outlet end of the UNM flume to catch insulation debris. The screen serves many purposes in the flume testing, but one of its important functions is to stop debris from recirculating or settling in the return sump of the laboratory. The model debris-catching screen is located 3 ft upstream from the outlet end of the flume. The flow-loss characteristics of the screen baffle are defined to give a 3-in.-H₂O pressure drop at a water velocity of 0.75 ft/s measured at the face of the screen. This behavior is consistent with effects observed in the flume.

Flow Source/Sink

A fluid source was specified in FLOW-3D that added water to the calculation at a location above water level, centered in the width of the flume, and 9-in. from the inlet end. The water from the source was routed down through a hollow object of revolution (inlet pipe) to a point either 27 in. or 12 in. above the floor of the flume, depending on the particular calculation. The inlet pipe was defined with an inside diameter of 6 in. These traits are similar to the orientation and size of the physical flow supplied to the UNM flume. The 27-in. and 12-in. inlet-pipe elevations correspond to the supply being introduced above the water surface or submerged, respectively.

A fluid sink was specified that removed water from the calculation at the identical rate at which the source added water. The sink was placed on the floor centered in the width of the flume 9 in. from the outlet end. The sink was circular, 10 in. in diameter, and 1½ in. high. These dimensions accurately represent the drain in the UNM flume.

Mass flow rates for the flow source and sink were specified at 202 gal./min. This flow relates to a uniform velocity in the body of the flume of 0.1 ft/s—a velocity of particular interest in the debris transport tests.

CFD Solution Options

Default solution schemes and convergence criteria were used in the FLOW-3D calculations. The free-surface logic was enabled to capture surface effects at the water/air interface. The fluid was treated as viscous with energy dissipation resulting from turbulence.

Boundary Conditions

No-slip, zero-velocity boundary conditions were defined on the bottom, ends, and sides of the model flume. A stagnant ambient pressure condition was defined at the top boundary.

Initial Conditions

An 18-in. water depth was defined for all calculations presented here. Velocity was initialized uniformly in the flume at 0.1 ft/s in the axial (lengthwise) direction. Pressure was initialized as hydrostatic.

UNM FLUME CONFIGURATIONS

CFD calculations were made for various flume configurations. The results for four configurations are presented here. They are characterized by the following.¹

1. All flow-smoothing structures near the inlet of the flume (diffuser and straighteners) in place with the exit of the flow-inlet pipe being above the water surface
2. The diffuser removed from the flume with the flow-inlet pipe elevated
3. The diffuser removed and the inlet pipe submerged
4. All flow-smoothing structures in place and a 3-in.-high curb placed just in front of the debris-catching screen

Each of the calculations was run for 15 min of real time. Typical CPU time for a calculation was 24+ h on a 450-MHz PC. Fifteen real-time minutes proved to be sufficient to establish steady-state conditions as indicated by unchanging average kinetic energy in the flow field and by unvarying velocity distributions.

RESULTS

The figures that follow show the results of the CFD simulations made of the UNM flume and tank experiments.

Flume

Results for the first flume configuration (defined above) are included. The figures show the three-dimensional (3D) velocity magnitude as a function of location. The coordinate system of the CFD model aligns the X-axis along the length of the flume, the Y-axis along the width of the flume, and the Z-axis along the height of the flume. The origin is at a lower corner of the flume where water is introduced. The positive directions of X and Z are down and up the flume, respectively. The positive direction of the Y-axis is consistent with a right-hand coordinate system. The CFD model of the UNM flume was built in SI units, so velocities in the figures are given in meters per second and lengths are given in meters. Different types of figures are included. Below are descriptions of each type.

Full-Flume Color-Scaled Figures

(The spatial extents of these figures match the full extents of the flume.)

- 3D velocity magnitude in a vertical length-height (X-Z) section located at mid-width (mid-Y) in the flume.
- 3D velocity magnitude in a horizontal length-width (X-Y) section located just above the floor of the flume.

Test-Region Color-Scaled Figures

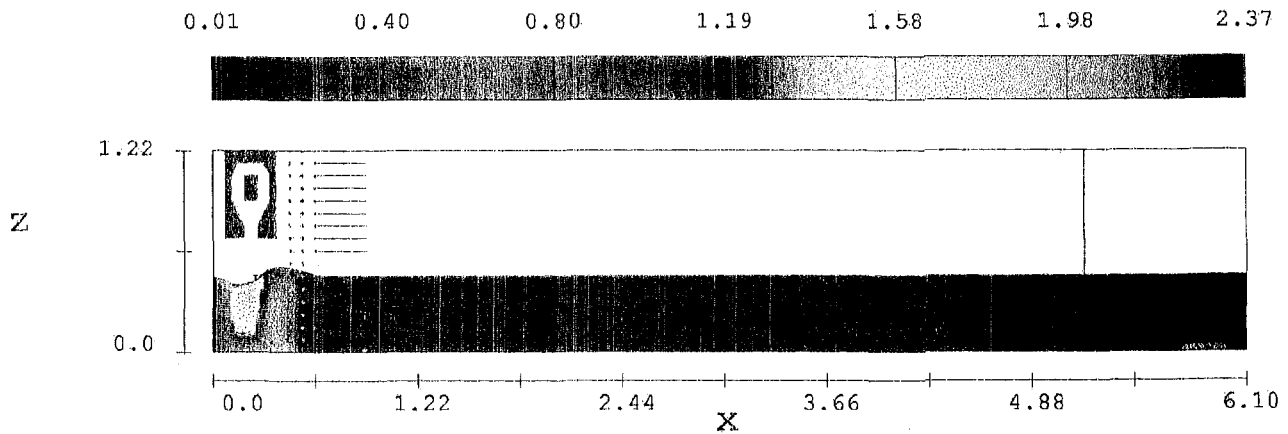
(The 10-ft length of the flume beginning where the flow straighteners end is referred to here as the "test region.")

- 3D velocity magnitude in a vertical length-height (X-Z) section located at mid-width (mid-Y) in the flume.
- 3D velocity magnitude in a horizontal length-width (X-Y) section located just above the floor of the flume. This section extends the full width of the flume.
- 3D velocity magnitude in a vertical width-height (Y-Z) section located centrally in the test region. This section extends the full width and height of the flume.

Tank

The first figure included for the tank CFD calculations shows the computer-aided drafting (CAD) input developed to describe the structure inside the 14-ft-diam tank. This model is supplied directly to FLOW-3D, which automatically maps a computational grid over the objects. The second figure is an elevated horizontal section identifying where falling water was introduced to the tank. (water was drained from the tank at a location diagonally opposite.) The third figure shows velocity vectors in a portion of the tank floor as the tank is filled. The fourth figure shows the 3D velocity magnitude on the floor at steady state (inflow matching outflow at a 1-ft water depth and 120 gal./min). The final figure illustrates vertical components of velocity in a portion of the tank, which represent an ability to suspend fine debris during transport. All calculations have been observed qualitatively in actual tests.

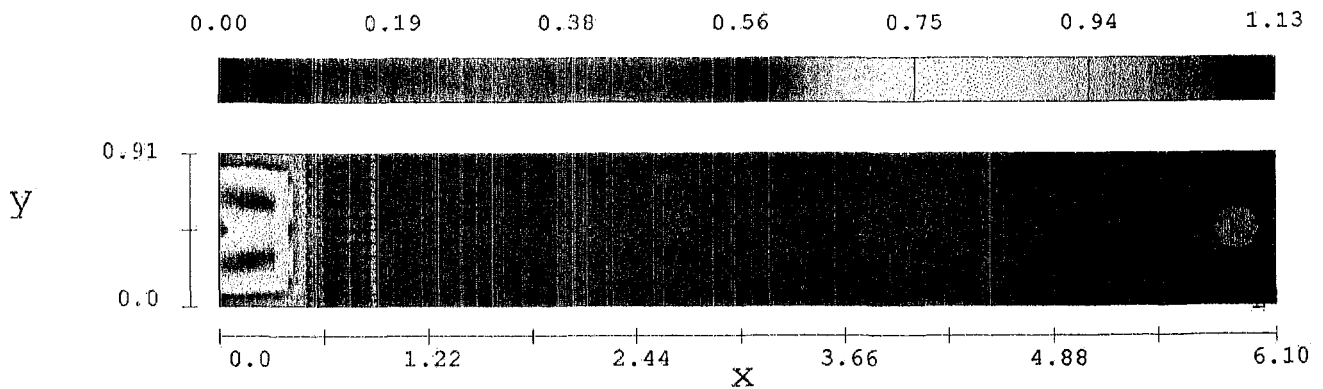
velocity magnitude contours



FLOW-3D® t=900.0 y=4.953E-01 (ix=2 to 82 kz=2 to 18)
 09:36:04 8-25-2000eifd hydr3d: version 7.6 win32 1999
 UNM flume

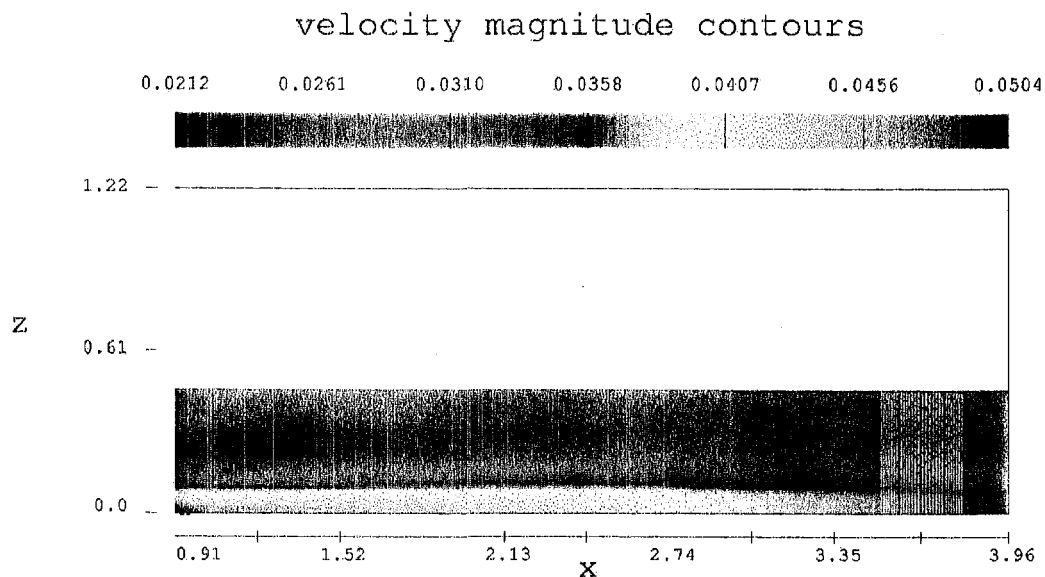
Full flume velocities (height-by-length section at mid-width).

velocity magnitude contours



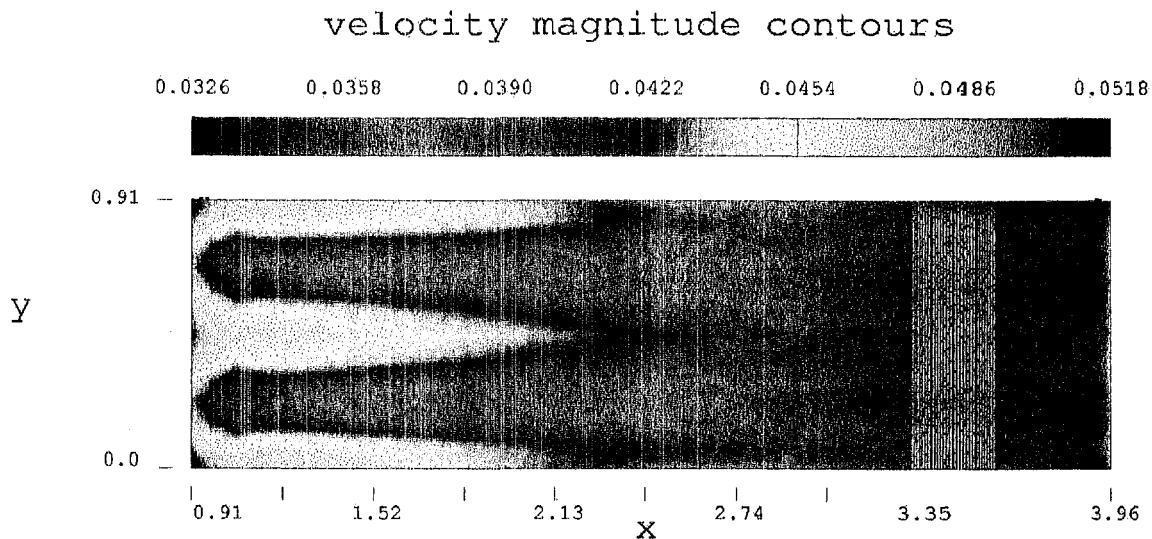
FLOW-3D® t=900.0 z=1.270E-02 (ix=2 to 82 jy=2 to 15)
 09:36:04 8-25-2000eifd hydr3d: version 7.6 win32 1999
 UNM flume

Full flume velocities (width-by-length section near floor).



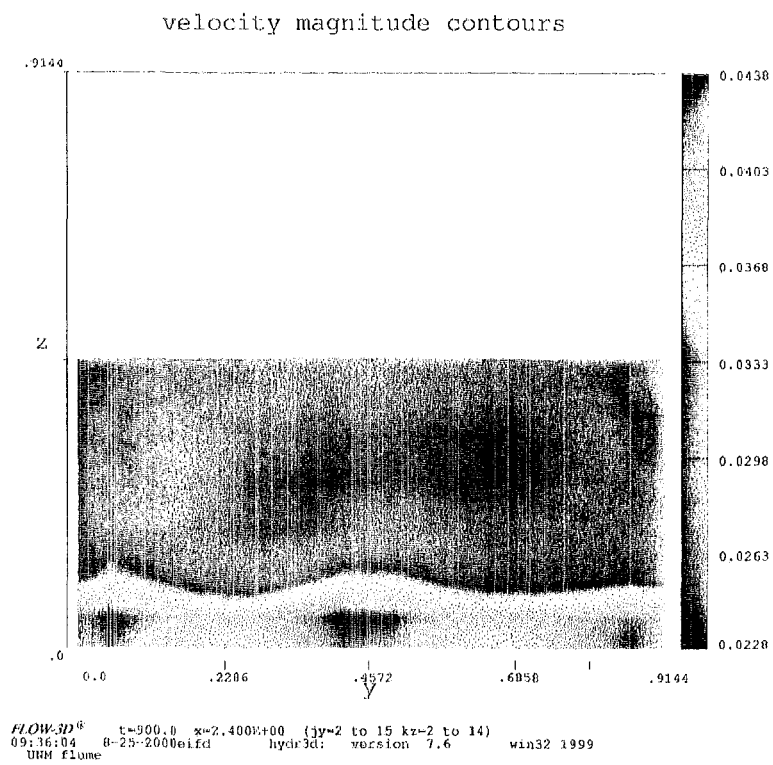
FLOW-3D® t=900.0 y=4.953E-01 (ix=15 to 54 kz=2 to 18)
 09:36:04 8-25-2000eifd hydr3d: version 7.6 win32 1999
 UNM flume

Flume test region velocities (height-by-length section at mid-width).

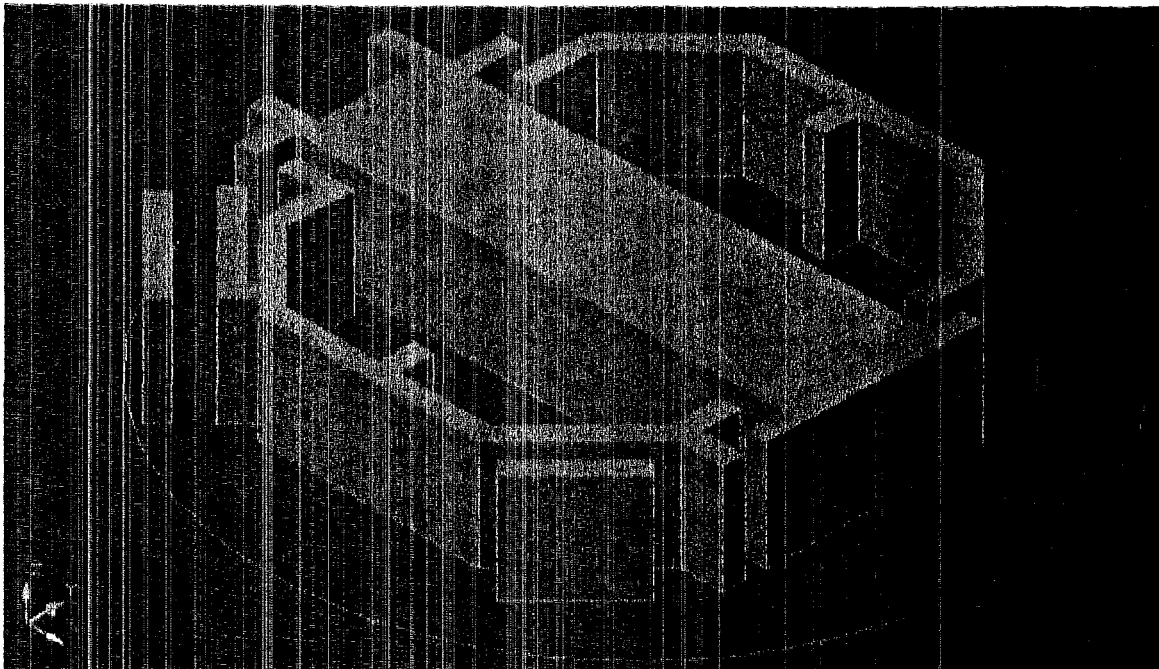


FLOW-3D® t=900.0 z=1.270E-02 (ix=15 to 54 jy=2 to 15)
 09:36:04 8-25-2000eifd hydr3d: version 7.6 win32 1999
 UNM flume

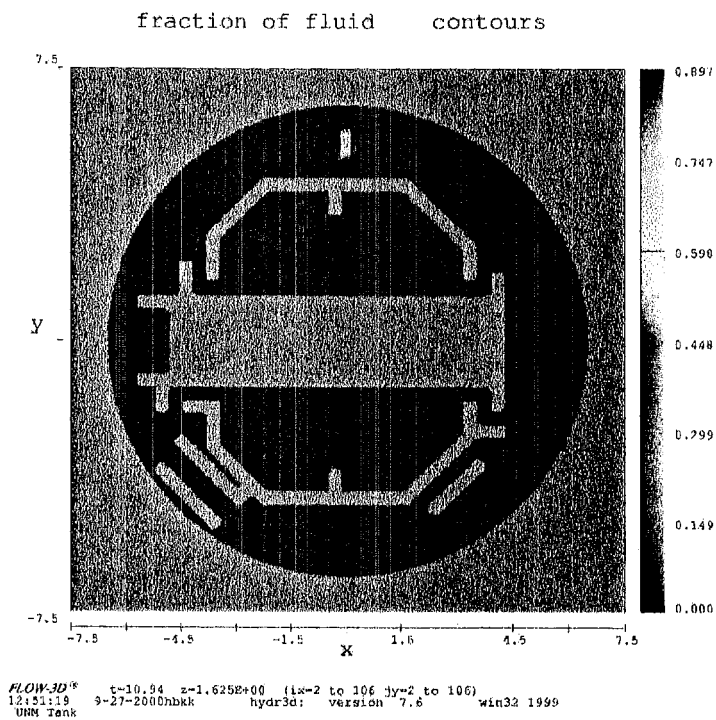
Flume test region velocities (width-by-length section near floor).



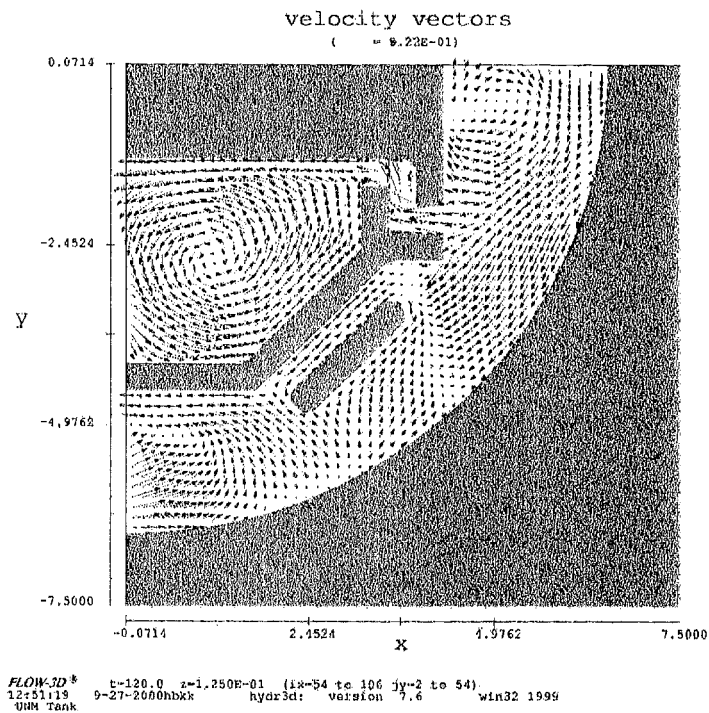
Flume test region velocities (width-by-height section at mid-length).



CAD model of obstacles in the UNM tank.

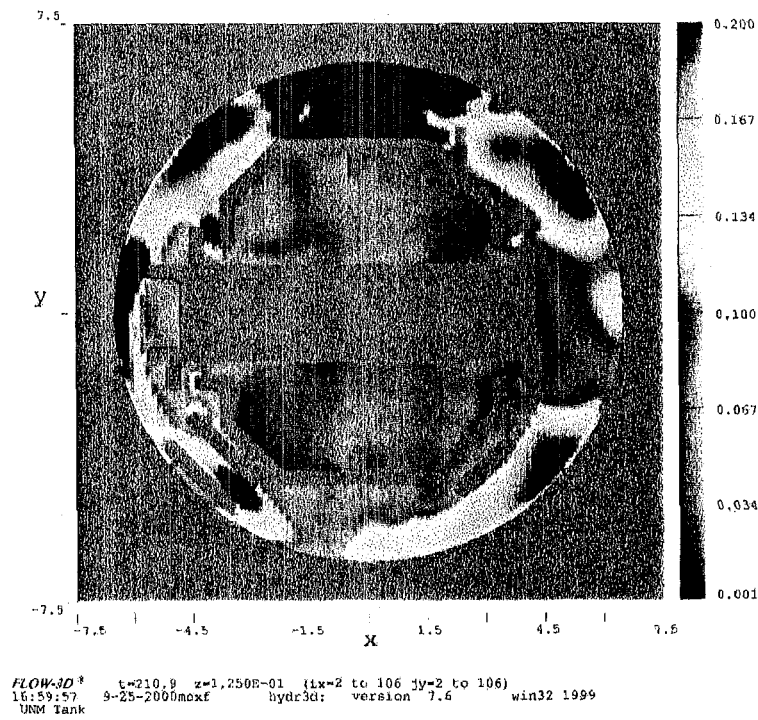


Location of the water source for all tank simulations.



Eddies in an interior cavity during filling.

velocity magnitude contours



Transport velocities near the tank floor.

velocity magnitude and vectors (vmax=2.91E-01)

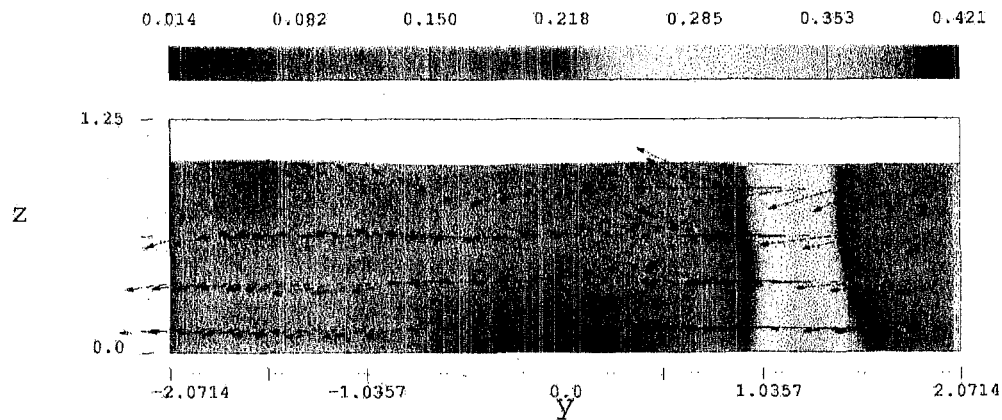


Illustration of lifting velocities in the tank.